HEAT TRANSFER APPLICATIONS IN TURBOMACHINERY

OF COnference 2007
2007 OpenFOAM Conference
November, 26-27, 2007 - London United Kingdom

HEAT TRANSFER APPLICATIONS IN TURBOMACHINERY

L. Mangani, C. Bianchini
Department of Energy Engineering ”Sergio Stecco”
University of Florence - Italy
Via Santa Marta, 3 - 50139 Florence - Italy
luca.mangani@htc.de.unifi.it

SUMMARY

This paper describes the development and validation steps needed to improve OpenFOAM standard release suitability for gas turbine heat transfer applications. The focus is on steady-state simulations as at the state of the art RANS is still considered a valid CFD approach especially for cases of industrial interest.

The most critical aspects were defined to be: the implementation of a steady state solver able at solving different kind of flow regimes, from almost incompressible to high Mach flows, and the turbulence modeling. A SIMPLE like algorithm was specifically developed to solve the fully three dimensional, steady state form of compressible Navier Stokes equations. Moreover a set of various eddy viscosity models, including several Low-Reynolds k-epsilon models also with realizability constraint and the Two Layer k-epsilon model were implemented. Due to the good performances obtained with the k-omega SST in wall-bounded flows using a Low Reynolds approach, an automatic wall treatment switching automatically form a wall-function to a Low-Reynolds formulation has been added for such model to obtain mesh independence also at high y_p^+. In addiction an anisotropic model, doping lateral diffusion of turbulence, was coded to better perform in plane film cooling and effusion cooling tests. The accuracy of the implementations was validated comparing results with experimental data available both from standard literature test cases and from in-house performed experiments. The geometries considered as validation tests cover the typical heat transfer problems in gas turbine design, namely impingement jets, film cooling and effusion cooling. During the tests, OpenFOAM code has shown a good accuracy and robustness as well as a remarkable computational speed.

The purpose of this paper is to present the results of the work done to test OF as an effective substitute for standard commercial CFD packages, both for academic and industrial users, in the specific field of heat transfer applications in turbomachinery.

INTRODUCTION

It is known how the study of cooling devices for turbomachinery is nowadays a keypoint in the design of both aeronautic and heavy duty engines. This is due to the great influence such systems have on the entire machine behavior both in terms of enhancing life of hot components, increasing the overall efficiency of the system and reducing exhaust emissions.

Accurate heat transfer measurements are however very expensive to support due to the complexity of geometries, the high costs of measuring device and the long set up necessary to collect reliable data. CFD analysis is as a consequence becoming more and more popular in each phase of the design process. Nevertheless, the numerical evaluation of thermal loads and effectiveness of the cooling devices in gas turbine engines is one of the most complex to face.

First of all the intricate geometries of reference, such as ribbed or pin-finned internal blade ducts, require quite flexible mesh criteria and the capability of solving for hybrid unstructured meshes become a must. In addiction the importance of well predicting the flow behavior in the vicinity of solid walls imposes a very fine spacial discretization increasing a lot grid dimensions and complexity. Moreover standard cooling systems involve many different kinds of flow, each one with is own peculiarity and characteristic, meaning that also the physical modeling will differ case by case. Typical example is the treatment of turbulence for which the use of specific advanced models is a keypoint in the success of the numerical predictions. A large set of turbulence models is so necessary not to limit correct simulation to a specific class of flows.// All these requirements have up to now limited the choice of CFD codes to few well-known commercial codes, for heat transfer analysis in turbomachinery both in academic and industrial field. Even though these codes offers numerous advantages, amongst which the ease of use and the large amount of built-in models, expert users may find several big drawbacks. Being thought for being multipurpose solvers in fact, their performances in terms of computational time and allocating resources result very poor. Due to their fixed and hidden internal structure, the possibility to introduce in the code user-defined models is usually limited, time-consuming and not very efficient as the complexity of the model grows. This is a very limiting issue both for academic and industrial $R\&D$ departments who need to manipulate the code to introduce "ad hoc" models often tuned on specifically conducted experiments. A code like OpenFOAM is indeed more suited for this type of user. Anyhow the lack of a steady-state compressible solver and of advanced turbulence models for heat transfer phenomena of the official release, limit a lot the reliability of the numerical predictions for this specific case.// The aim of the present paper is so to show the developments introduced in OpenFOAM standard release to make it suitable for steady state heat transfer analysis able to assist gas turbine design process. A fully compressible steady state pressure-based solver has been introduced together with a set of turbulence two-equations closures with particular reference to a detailed near wall treatment. As confirmation of the work done a set of validation testcases were performed. In particular, in this paper, we will focus our attention on the validation of the code with some complex configurations typical of heat transfer problems such as film cooling and impingement cooling. Both film and impingement cases were analyzed with single and multi-hole configurations. Comparisons with experimental data are reported in terms of adiabatic effectiveness for film cooling tests and wall heat transfer coefficient for impingement runs. Furthermore, in order to verify the accurate implementation of selected turbulence models, a simple flat plate tests was considered, while to show the robustness of the steady state solver developed results of classical validation tests are briefly commented.

NOMENCLATURE

U	Vector velocity	$[m\ s^{-1}]$
D	Hole diameter	$[m]$
h	Heat transfer coefficient	$[W\ m^{-2}\ K^{-1}]$
k	Turbulent kinetic energy	$[m^2\ s^{-2}]$
L	Height of the jet	$[m]$
p	Pressure	$[N\ m^{-2}]$
p'	Pressure corrector	$[N\ m^{-2}]$
$\dot{q}$	Heat flux	$[W\ m^{-2}]$
Re	Reynolds number	
T	Temperature	$[K]$
T_s	Turbulent time scale	$[s]$
P_k	Turbulence production term	
	$= \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial U_k}{\partial x_k}\right)\frac{\partial U_i}{\partial x_j} - \frac{2}{3}\rho k \frac{\partial U_j}{\partial x_j}$	$[kg\ m^{-1}\ s^{-3}]$
S	Tensor strain $= 0.5\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)$	$[s^{-1}]$
X	Streamwise direction	$[m]$
Y	Spanwise direction	$[m]$
C_ρ	Compressibility term $\frac{1}{RT}$	$[kg\ J^{-1}]$
H	Diffusive discretization term	$[kg^{-1}\ s\ m^3]$

Greeks

α	Angle between hole and crossflow	
η	Adiabatic effectiveness $\frac{(T_\infty - T_{aw})}{(T_\infty - T_c)}$	
$< \eta >$	Spanwise averaged effectiveness $\sum_n \frac{(T_\infty - T_{aw})}{(T_\infty - T_c)}$	
ω	Turbulence frequency	$[s^{-1}]$
ε	Turbulence dissipation	$[m^2\ s^{-3}]$
$\mu_{t,eff}$	Eddy viscosity	$[kg\ m^{-1}\ s^{-1}]$
ρ	Density	$[kg\ m^{-3}]$
γ	Anisotropic factor	
Γ	Blending function	

Subscripts

0	Uncooled plate
aw	Adiabatic wall
∞	Crossflow
c	Coolant
w	Wall
awt	Automatic Wall Treatment
p	First near wall node
HR	High Reynolds
LR	Low Reynolds

SOLVER

In turbomachinery and heat transfer applications, involved fluid flows may cover a wide range of Mach regimes. In particular, it usually happens that different Mach conditions simultaneously arise in the same domain. Such situation makes the accurate solution of viscous flows governing equations a complex task.

To avoid the known weaknesses of density-based algorithms, another class of methods, proposed originally for viscous incompressible flows [1, 2, 3, 4] and later extended to compressible flows [5, 6, 7, 8, 9, 10, 11] use pressure as the main independent variable also with the concept of the 'retarded density' [12, 13, 14]. Such pressure-based approach is founded on the SIMPLE algorithm (Semi-IMplicit Pressure Linked Equations) [1]. In this method, continuity equation is converted into an equation for pressure corrector overturning the linkage between pressure and density to extend applicability range up to zero Mach number. The SIMPLE algorithm uses a segregated approach where the equations are solved in sequential steps letting to the iterative process the care of the non-linearity as well as the coupling between equations. To better visualize the cycle of SIMPLE algorithm a flow-chart of the pseudo code is reported in Fig. 1.

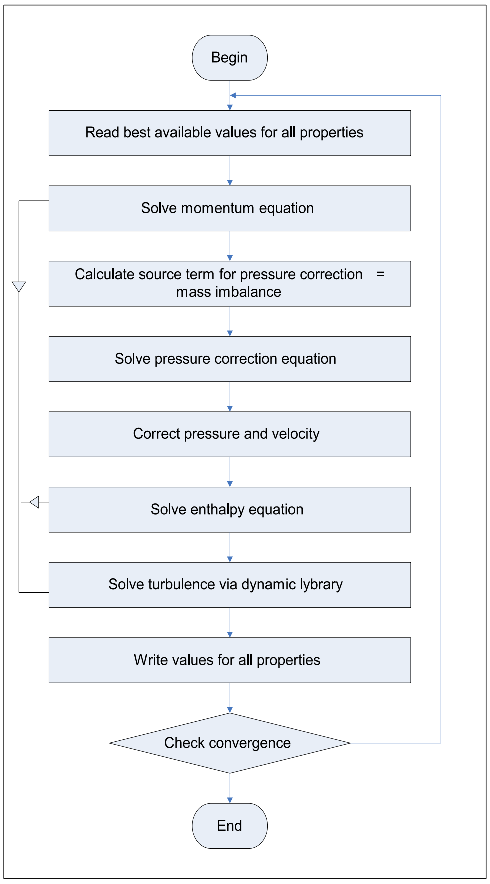


Figure 1. Flow chart of SIMPLE algorithm.

The improvements of this class of methods in contrast with the standard SIMPLE technique lies in a more precise derivation of the equation for the pressure corrector, allowing the possibility of treating at the same time low subsonic, almost incompressible, and high compressible flows.

The pressure correction equation, details can be found in [15, 16, 17, 18], in the compressible form says:

$$\nabla \cdot (C_\rho U p^{'}) - \nabla \cdot (\rho H (\nabla p^{'})) = -\nabla \cdot (\rho U). \qquad (1)$$

The role of Eq.(1) in the SIMPLE cycle is to enforce mass conservation, it is in fact derived from a combination of momentum conservation and continuity equation. In order to solve Eq.(1), attention should be posed on the fact that the pressure correction equation now assumes a convective-diffusive form instead of a purely diffusive behavior like the original incompressible formulation. While the other steady-state form transport equations have to be relaxed in order to characterize the inertial physics lost by the elimination of the time derivative, for the pressure correction equation this cannot be done. Usage of usual implicit relaxation techniques on pressure corrector, in fact, corrupt mass conservation on single iteration steps breaking the concept standing behind SIMPLE algorithm. In subsonic cases, standard Neumann conditions at inlet velocity boundary, like in incompressible tests, determine ill-defined problems for Eq.(1). Care must be taken in handling pressure correction boundary condition in order to solve in a well-posed manner such an equation [19]. A combination of Dirichlet and Neumann type condition for the inlet has been tested.

TURBULENCE MODELS

The correct modeling of turbulent quantities is fundamental in conducting heat transfer simulations, because of the simultaneous importance of well predicting both the near wall behavior and the complex structures of the main flow [20, 21]. Correct predictions of thermal quantities and gradients inside boundary layers are necessary to establish whether or not the cooling systems of gas turbine are efficient. At the same time wall properties are very dependent on the development of the free stream flow.

Use of standard wall function has to be avoided because of the unpredictability of boundary thermal gradient and the failure in predicting transitional, Low Reynolds as well as adverse pressure gradient flows.

First step in modeling turbulence in a more appropriate manner inside the near wall region has been the introduction of the so-called damping functions in the standard $k - \varepsilon$. The basic structure of the models is the same for all of them differing in the tuning of the damping functions and some extra sources in dissipation equation, as shown below:

$$\frac{\partial \rho k}{\partial t} + \nabla \cdot (\rho U k) - \nabla \cdot (\mu_{eff} \nabla k) = P_k - \rho\varepsilon \; , \qquad (2)$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \nabla \cdot (\rho U \varepsilon) - \nabla \cdot (\mu_{eff} \nabla \varepsilon) = f_1 P_k \frac{\varepsilon}{k} - f_2 \rho \frac{\varepsilon^2}{k} \; . \qquad (3)$$

Of the many Low Reynolds $k-\varepsilon$ models proposed in literature in the course of years, the models by Lien and Leschziner [12], Lien [22], Abe *et al* [23], Chien [24], Chen *et al* [25], Hwang and Lin [26] and Lam and Bremhorst [27] have been implemented.

It is however known in literature that in high strain rate regions eddy viscosity models tend to overpredict turbulent kinetic energy: this problem is sometimes referred to as “stagnation point anomaly” [28]. These higher values of k are due to an overestimate of production term P_k. To avoid such overprediction linear dependence between P_k and $|S|^2$ should be bounded in regions where $|S|$ grows. This is achieved with a time scale bound, derived from a "realizability" constraint for Reynolds stress tensor to be definite positive:

$$T_s = \frac{\mu_t}{C_\mu \rho k} = min\left(\frac{k}{\varepsilon}, \frac{\alpha}{\sqrt{6} C_\mu |S|}\right) \quad (4)$$

This limiter proposed by Durbin [29] has been inserted in all Low Reynolds models above presented as an option to be switched on or off by the user.

Then, in order to match good near wall predictions with suitable modeling of flow structures far from the wall, Two Layer $k-\varepsilon$ models have been implemented. Such method consists in patching together a one equation model in the near wall layer and a two equation High Reynolds model in the outer layer [30]. Both Wolfestein and Norris&Reynolds closure formulas [22] have been tried without significant discrepancies in the results.

Another failure of numerical computations, already pointed out in previous studies [31, 32], is the low lateral spreading predicted for jets in crossflow. Both film and effusion cooling geometries tend in fact to concentrate the cooling effect on the meanline of the holes. This is due to the assumption of turbulence isotropy that fails in the near-wall region because of the damping of normal to wall fluctuations. To avoid this discrepancy with experimental evidences, Bergeles [33] proposed to algebraically correct the main Reynolds stress, enlarging the product of the stream and span directed fluctuation. The Reynolds Stress tensor is so calculated using a tensorial definition of turbulent viscosity:

$$-\rho\overline{u_i^{'} u_j^{'}} = \mu_{t,ij}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial U_k}{\partial x_k}\right), \quad (5)$$

where

$$\mu_{t,ij} = \begin{pmatrix} \mu_t & \gamma\mu_t & \mu_t \\ \gamma\mu_t & \mu_t & \mu_t \\ \mu_t & \mu_t & \mu_t \end{pmatrix}, \quad (6)$$

and γ is an amplification factor ranging from 4.5 in the freestream region to 60 in the near wall region (the model is applied for $y^+ > 1.5$) [34].

This correction was applied to the Two Layermodel changing the turbulent flux in the momentum, energy and turbulence equations.

All previous models require a quite fine grid at walls ($y^+ \approx 1$) but for cases of a certain complexity this results to be a quite strict constraint. In order to increase grid independence a mixed approach between wall-function and Low Reynolds was added to the $k-\omega$ SST model. The idea, see [35], is to blend the two approaches via a blending function Γ calculated algebraically from the non dimensional wall distance. Both turbulent production and turbulent specific dissipation are imposed on the first node mixing the effects of the Low Reynoldsand the High Reynolds contributions:

$$P_{awt,p} = P_{LR,p} e^{-\Gamma} + P_{HR,p} e^{-\frac{1}{\Gamma}}, \quad (7)$$

$$\omega_{awt,p} = \omega_{LR,p} e^{-\Gamma} + \omega_{HR,p} e^{-\frac{1}{\Gamma}}, \quad (8)$$

where Γ is calculated with an algebraic expression for y^+ defined with $u_\tau = max(u_{\tau,LR}, u_{\tau,HR})$. The same blending is applied to thermal quantities following Kader universal law [36].

RESULTS

Generalities

Most of the cases to be presented have already been tested and published in [37] where much more details on the cases can be found. Even if many of the already obtained results are reported to make comparisons, here the focus will be on the new models to be tested: the anisotropic model and the automatic wall treatment for the $k-\omega$ SST.

Due to the great number of implemented turbulence models, a shortcut has been used to name most of them: the acronyms presented in Tab. 1 will be widely used in substitution of authors' full name.

Table 1. Acronyms for the various turbulence models.

$k-\varepsilon$ Low Reynolds by Abe et al.	AKN
$k-\varepsilon$ Low Reynolds by Chien	CH
$k-\varepsilon$ Low Reynolds by Lien et al.	CLL
$k-\varepsilon$ Low Reynolds by Hwang and Lin	HW
$k-\varepsilon$ Low Reynolds by Lam and Bremhorst	LB
$k-\varepsilon$ Low Reynolds by Lien and Leschziner	LW
$k-\varepsilon$ Low Reynolds by Lien	LNR
Realizability constraint correction	Real
Two Layer	TL
Anisotropic Two Layer	KEA
Automatic Wall Treatment	SSTawt

Turbulence Models Validation Test

Flat Plate

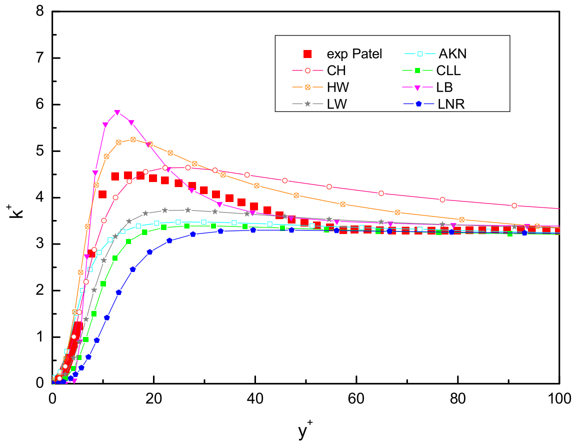
First test to be presented is a fully turbulent flow over an adiabatic flat plate. Such simple case was chosen in order to test in detail the near wall behavior of turbulence, that is why a very fine grid was needed. The flow field being modeled is that reported by Wieghardt [39] and later included in the 1968 AFOSR-IFP Stanford Conference [40]. Details about flow conditions are listed in Tab. 2.

Table 2. Flow conditions for flat plate test

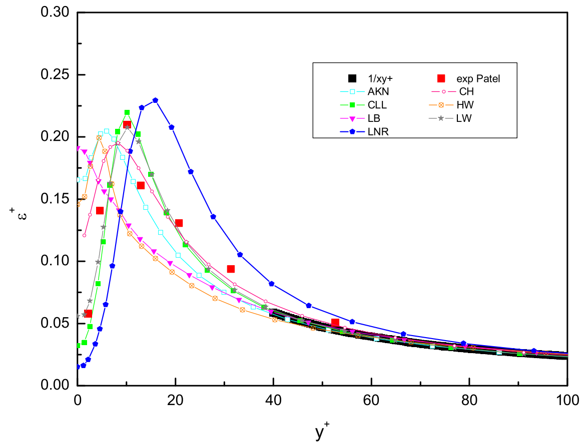
Inlet temperature	294.4	K
Inlet Mach number	0.2	
Pressure	101400	Pa
Turbulence kinetic energy - k	23.6	m^2s^{-2}
Dissipation - ε	3365	m^2s^{-3}
Comparison axial loc. - x	4.6870	m

Both the non dimensional k and ε were checked in their behavior against the non dimensional wall distance at an axial location where the flow can be considered fully developed, Fig. 2. The level of agreement with the experimental is fairly good, especially for k^+, with free stream values matching for almost all models. However the discrepancy between experimental and numerical simulation follows exactly the profiles obtained by other researchers some of which are directly models' authors [41, 24, 26]. Both for a higher guarantee of stability and for the best reliability on matching experimental results further test was conducted only on CLL and AKN.

For the same geometry another case involving heat transfer was set up to validate mesh independence of the $k-\omega$SST with the automatic wall treatment. Three different grids with increasing first node distance from the wall were tested ($y_p^+ = 0.1; y_p^+ = 9; y_p^+ = 35$). The results are reported in terms of Stanton number and tested against Incropera [42], see Fig.3. Even if the three profiles are still not coincident the mesh dependence is decreasing as the flow develops and moreover the agreement is improved a lot from standard wall function approach.



(a) k^+ profile



(b) ε^+ profile

Figure 2. Turbulence quantities profiles.

Heat Transfer tests

Impingement Cooling

As already known in literature, impinging jets flows are a specific class of flows for which the modeling of turbulence results determinant in obtaining acceptable agreement with experimental data. Especially for flows involving heat transfer in fact, the flow conditions in the area around the stagnation point result pretty dependent on the turbulence model used. The numerical estimation of heat transfer coefficient, for example, can lead to errors of about 100% in the evaluation of the peak, choosing one or another model. This class of flows is so suitable for deeply testing new turbulence models.

Two different impinging geometries were tested: first of all a 2-D normal impinging jet of air has been performed following a test case by ERCOFTAC and after a five row array of holes reproducing experimental results obtained during the European project LOPOCOTEP.

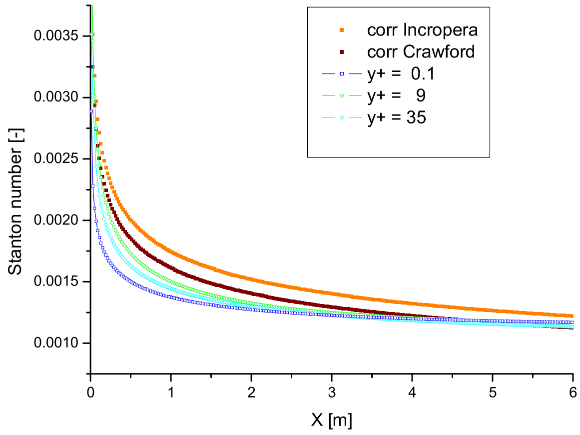


Figure 3. Stanton number profile.

ERCOFTAC C25 Axial-symmetric Impingement This case is a turbulent incompressible flow of air impinging onto a flat plate, modeled after [43]. The impact surface is heated and kept at constant heat flux of $\dot{q} = 200\, W/m^2$, all other walls were treated as adiabatic walls. From the experiments, the Nusselt number distribution for various jet Reynolds numbers is known. For validation purpose, a Reynolds number of 23.000 and a distance of $L/D = 2$ were chosen. The far-field boundaries are modeled as mixed inflow/outflow pressure boundaries.

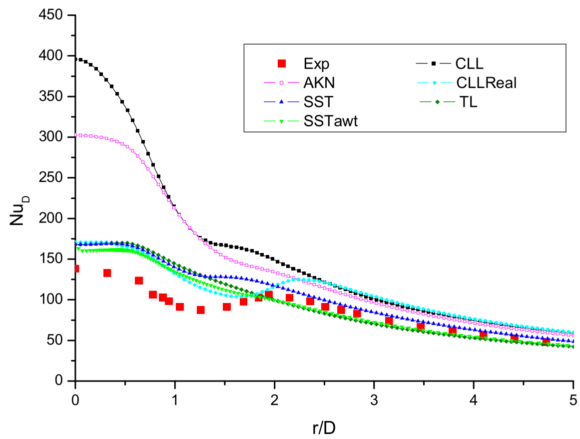


Figure 4. Nusselt number distribution along radius.

Fig. 4 shows how the models without realizability constraint fail and dramatically overpredict the peak in the heat transfer coefficient (error of almost 200%) in agreement with the original idea of such limiter [44].

In addition to what already noticed in [37], it is possible to compare the behavior of the SSTawt model with the other models. When the same Low Reynolds mesh is used its predictions are obviously collapsing on the standard SST model and agreement with the best performing models is good.

5-Holes Impingement cooling This case is simulating typical design conditions for impingement cooling of a gas turbine. The case was performed following the set up of an experiment done at the Energy Engineering Department of the University of Florence for the European project LOPOCOTEP (LOw POllutant COmbustor TEchnical Programme). Coolant is injected from a plenum through a perforated plate and impacts over a flat plate at uniform heat flux reproducing the behavior of the first $3-2$ rows (the array of holes is staggered) for a total of 5 jets. For further details refer to [45].

Main flow parameters are reported in Tab. 3.

Table 3. Flow conditions for 5-hole impingement test

Inlet Temperature	308.2	K
Outlet Pressure	85101	Pa
Inlet Turbulence level - Tu	$\leq 0.5\%$	%
Re_j	7600	
Inlet Velocity	0.28956	m/s
Wall Heat flux	3000	W/m^2

Simulations have been validated in terms of heat transfer coefficient, see Fig. 5 and Fig. 6, calculated with respect to inlet static temperature, almost coincident for such low Mach number with inlet total temperature. Adiabatic simulations have been conducted too but on a single hole configuration of the same case, in order to check whether this approximation could be done or not.

Both experimental and numerical data are sampled onto the two different lines connecting symmetry planes and then merged together in the zone where a relative minimum is localized.

Even if obtained results are in good agreement with experimental data far from the stagnation point, it should be noticed that predictions for the peak value are quite different from measured data. Higher discrepancies on the even peaks are probably due to errors in the experimental measurements [45]. Comparing the two models, Two Layer predicts peak values a 10% better of the SST giving basically identical results outside the stagnation points area. In any case, it should be considered that temperature gradients are quite small. A better agreement is expected for higher values of wall heat flux.

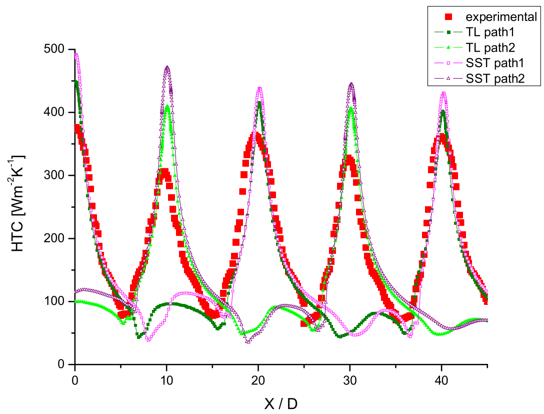
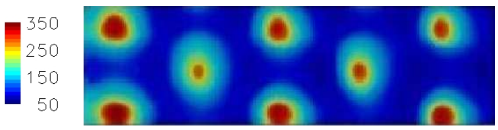
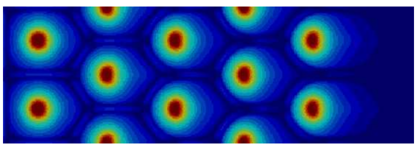


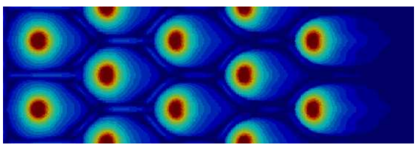
Figure 5. Heat transfer coefficient along center lines.



(a) Experimental



(b) Two Layer



(c) k-ω SST

Figure 6. Heat transfer coefficient $[W\ m^{-2}\ K^{-1}]$ distribution on impinged wall.

Film and effusion cooling

This section is presenting results for jets in crossflow aimed at representing both film and effusion cooling. The main difference in these two methods of cooling stands actually just in which is the principal cooling effect of the cold flow injected in the hot gases. In film cooling the heat absorbed in the small hole is negligible and wall is protected with a film of coolant over the external surface. For effusion cooling viceversa the less effective film protection is compensated by a higher heat removed in the holes.

The numerical prediction of the mixing between coolant and cross flow still represents one of the most difficult task in CFD analysis, even in adiabatic conditions [46]. The difficulties are mainly related to the well known deficiency of standard eddy viscosity turbulence models in the accurate prediction of lateral jet spreading, essentially due to the isotropy assumption for turbulent stresses [47, 48, 49]. The performances of the KEA model were so tested against standard EVM in terms of averaged and local adiabatic effectiveness.

Two different test-cases were studied: the well known single hole experiment by Sinha [50] and an experimental multi-hole geometry aimed at cooling turbine endwalls [51].

Sinha test Experimental data and geometries are based on tests made by Sinha *et al.*[50]; local and spanwise averaged effectiveness are compared with calculated values. The geometry is a flat plate with a single row of holes, while flow conditions are listed in table 4.

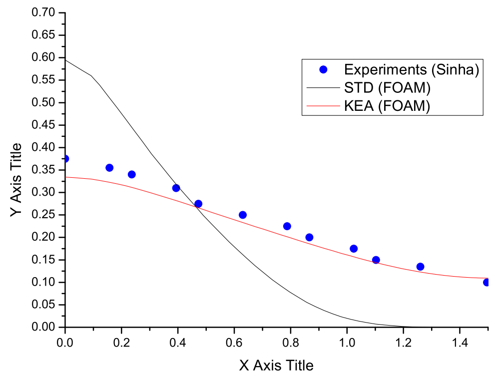
Table 4. Flow conditions for Sinha test

Cross flow temperature	300	K
Coolant temperature	153	K
Pressure	10^5	Pa
Density ratio - DR	2.0	
Blowing-rate - M	0.5	
Momentum ratio - I	0.125	
Turbulence level - Tu	≤ 0.2	%
Cross flow velocity	20	m/s
Re_c	15700	

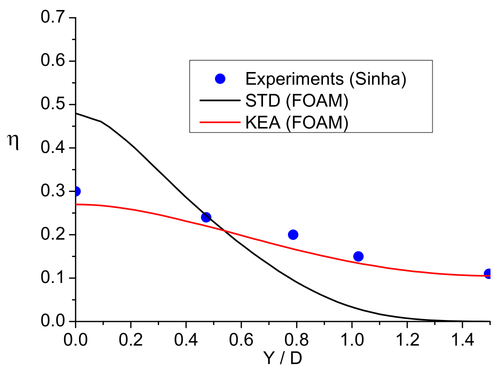
The performances of standard turbulence models (STD model in Figs.7(a) and 7(b) were analyzed in comparison with the anisotropic model (KEA) in terms of laterally-averaged and local span-wise and centerline effectiveness.

Local lateral effectiveness 10 and 15 diameters downstream is shown in Fig. 7, local centerline and laterally averaged effectiveness is also presented in Fig. 8. Finally a map of wall effectiveness is reported in Fig. 9.

The improvements obtained using the anisotropic model are remarkable. The experimental lateral profiles of effectiveness are perfectly reproduced both at 10 and 15 diameter downstream while all the other isotropic models where failing such predictions dramatically. The centerline values also tend to be much closer to experiments especially when the distance from the hole is increasing. It is clear how the eddy viscosity model predicts a too coherent jet Fig. 9, thus severally underestimating lateral cooling performances. However this error is smoothed in the laterally-averaged parameters and as expected the anisotropic factor just distribute in different way the coolant over the plate

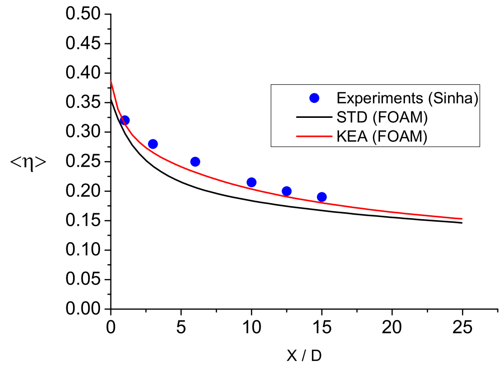


(a) 10D

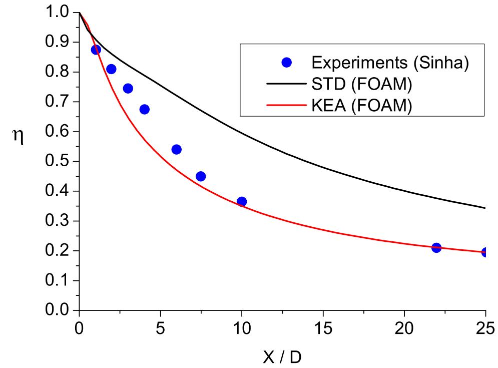


(b) 15D

Figure 7. Spanwise distribution of film cooling effectiveness at various sections.



(a) Laterally averaged



(b) Center line

Figure 8. Comparison between laterally averaged and local center line film cooling effectiveness.

not influencing that much the averaged effectiveness. This consideration allows the use of isotropic models when the averaged parameters are considered but as expected confirm how, the local values of wall temperature cannot be considered reliable.

6-Holes effusion cooling The geometry of this case is a six holes flat plate interposed in between a plenum and a channel at lower pressure. A summary of flow conditions can be found in Tab. 5.

Results are reported in terms of spanwise averaged adiabatic effectiveness, see Fig. 10. Together with experimental data, correlative approach predictions using L'Ecuyer and Soechting correlation with Sellers superposition criterion have been reported [51]. It is in fact quite difficult to reproduce exactly such experimental profiles that are better used as a reference mean value for η. In agreement to what already noticed in the previous case, the

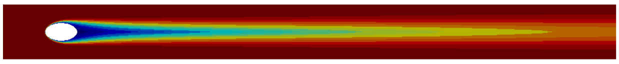
Table 5. Flow conditions for 6-holes effusion test

Cross flow temperature	323	K
Coolant temperature	298	K
Pressure	$7.0 \cdot 10^4$	Pa
Density ratio - DR	1.103	
Blowing-rate - M	0.2	

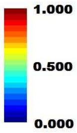
laterally averaged effectiveness is not that much influenced by the anisotropic factor, see Fig.10.

All models in fact qualitatively well reproduce the correlative decay of the spanwise averaged effectiveness downstream the holes. By looking at the two-dimensional effectiveness map in Fig. 11 however it's evident how the lateral diffusion of coolant is more correct in the anisotropic case than in the other two. For the lack of experimental data this analysis could only be done qualitatively.

(a) KEA



(b) STD



(c) Color map

Figure 9. Effectiveness distribution over the wall.

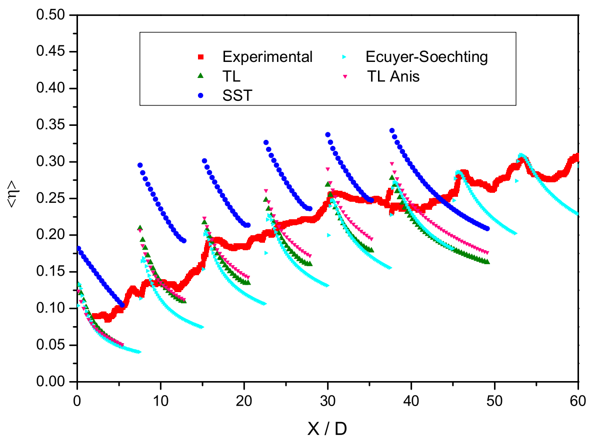


Figure 10. Spanwise averaged adiabatic effectiveness.

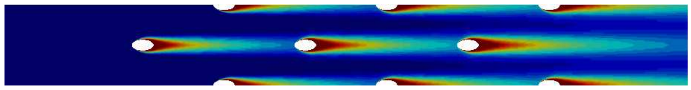
CONCLUSIONS

OpenFOAM has been improved to predict heat transfer phenomena in gas turbine. Many different tests were performed representing the state of the art for the cooling systems in turbomachinery applications. Validation of a pressure correction algorithm and various turbulence models have been made by comparison with experimental data on typical heat transfer geometries. Massive parallel calculation have also been tested for the multirow configuration simulations both for impingement and effusion cases by the use of LAM/MPI library `http://www.lam-mpi.org`.

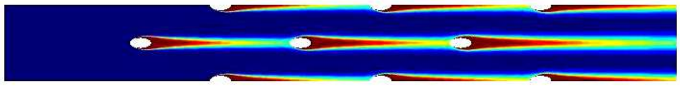
The combination of the new built-in OpenFOAM libraries is able to reproduce the flow conditions with good accuracy for all the geometries studied. Good agreement with experimental data and with the common commercial software has been reached for impingement and effusion cooling configurations.

The used object oriented language results to be very flexible for implementing new turbulence models, solver algorithms, boundary condition types and physical models.

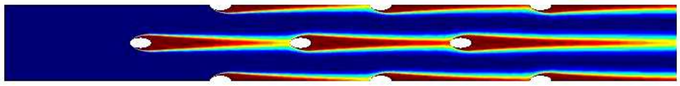
Future work will be concentrated on expanding the capability of the code to simulate fluid-structure interaction, with main focus in conjugate heat transfer analysis. An even more fundamental step will be to expand the capability of the code to handle periodic boundaries, in particular it is necessary to implement a new non-conformal type of cyclic patch.



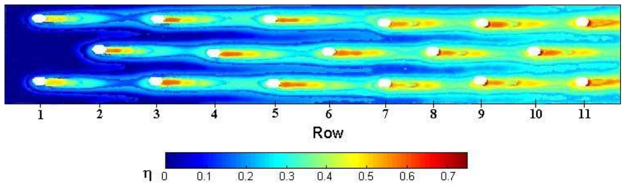
(a) Two LayerAnis



(b) Two Layer



(c) SST



(d) experimental

Figure 11. Comparison between laterally averaged and local center line film cooling effectiveness.

ACKNOWLEDGMENT

This work was conducted within a collaboration contract between th Energy Engineering Department of the University of Florence and the industrial partner Ansaldo Energia Spa.

References

[1] S. V. Patankar. *Numerical Heat Transfer and Fluid Flow*. Taylor & Francis, USA, 1980.

[2] W. Malasakera and H. K. Versteeg. *Computational Fluid Dynamics*. Longman Scientific, England, 1995.

[3] M. Peric. Numerical methods for computing turbulent flows. Technical Report VKI LS 2004-06.

[4] J. H. Ferziger and M. Peric. *Computational Methods for Fluid Dynamics*. Springer, Germany, 2002.

[5] M. Peric, Z. Lilek, and L. Demirdzic. A collocated finite volume method for predicting flows at all speed. *Journal of Numerical Methods in Fluids*, 16:1029–1050, 1993.

[6] M. Darbandi and G. E. Schneider. Application of an all-speed flow algorithm to heat transfer problems. *Numerical heat transfer*, 35:695–715, 1999.

[7] C. M. Rhie. Pressure based navier-stokes solver using the multigrid method. *AIAA Journal*, 27(8):1017–1018, 1989.

[8] W. Shyy and M. E. Braaten. Applications of a generalized pressure correction algorith for flows in complicated geometries. *Advances and Applications in Computational Fluid Dynamics - ASME Winter Annual Meeting*, pages 109–119, 1988.

[9] W. Shyy and M. E. Braaten. Adaptive grid computation for inviscid compressible flows using a pressure correction method. *AIAA, ASME, SIAM, and APS, National Fluid Dynamics Congress*, pages 112–120, 1988.

[10] K. C. Karki and S. V. Patankar. Pressure based calculation procedure for viscous flows at all speeds in arbitrary configurations. *AIAA Journal*, 27(9).

[11] J. Rincon and R. Elder. A high resolution pressure based method for compressible flow. *Journal of Computation and Physics*, 3:217–231, 1997.

[12] F. S. Lien and M. A. Leschziner. A pressure-velocity solution strategy for compressible flow and its application to shock/boundary-layer interaction using second-moment turbulence closure. *Journal of fluids engineering*, 115:717–725, 1993.

[13] E. S. Politis and K. C. Giannakoglou. A pressure-based algorithm for high speed turbomachinery flows. *International journal for numerical methods in fluids*, 25:63–80, 1996.

[14] J. J. McGuirk and G. Page. Shock capturing using a pressure-correction method. *AIAA Journal*, 28(10):1751–1757, 1990.

[15] F. Moukalled and M. Darwish. Tvd schemes for unstructured grids. *International Journal of heat and mass transfer*, 46:599–611, 2003.

[16] F. Moukalled and M. Darwish. Normalized variable and space formulation methodology for high-resolution schemes. *Numerical heat transfer. Part B, fundamentals*, 26:79–96, 1994.

[17] F. Moukalled and M. Darwish. A unified formulation of the segregated class of algorithms for fluid flow at all speeds. *Numerical heat transfer. Part B, fundamentals*, 37:103–139, 2000.

[18] F. Moukalled and M. Darwish. A robust pressure based algorithm for multiphase flow. *International journal for numerical methods in fluids*, 41:1221–1251, 2003.

[19] I. Senocak and W. Shyy. A pressure based method fot turbulent cavitating flow computations. *Journal of Computation and Physics*, 176:363–383, 2001.

[20] L. Davidson. An introduction to turbulence models. Technical Report Publication 97/2, Chalmers University of Technology, 2003.

[21] J. Bredberg. On two equation eddy-viscosity models. Technical Report Internal report 01/8, Chalmers University of Technology, 2001.

[22] F. S. Lien. *Computational modeling of* 3*-D flow in complex ducts and passages*. PhD thesis, University of Manchester, Institute of science and technology, 1992.

[23] K. Abe, T. Kondoh, and Y. Nagano. A new turbulence models for predicting fluid flow and heat transfer in separating and reattaching flows-i. flow field calculations. *International Journal of Heat Mass Transfer*, 37:139–151, 1994.

[24] D. A. Yoder and N. J. Georgiadis. Implementation and validation of the chien $k - \varepsilon$ turbulence model in the wind navier-stokes code. Technical Report 209080, NASA, Glenn Research Center, Cleveland Ohio, 1999.

[25] F. S. Lien, W. L. Chen, and M. A. Leshziner. Low-reynolds-number eddy-viscosity modelling based on non-linear stress-strain/vorticity relations. *Engineering turbulence modelling and experiments*, 3:91–100, 1996.

[26] C. B. Hwang and C. A. Lin. Improved low-reynolds-number $k - \varepsilon$ model based on direct numerical simulation data. *AIAA Journal*, 36(1):38–43, 1998.

[27] C. K. G. Lam and K. Bremhorst. A modified form of the $k-\varepsilon$ model for predicting wall turbulence. *Journal of fluids engineering*, 103:456–460, 1981.

[28] P. A. Durbin. On the $k-\varepsilon$ stagnation point anomaly. *International journal of heat and fluid flow*, 17(1):89–90, 1996.

[29] G. Medic and P. A. Durbin. Towards improved prediction of heat transfer on turbine blades. *Journal of turbomachinery*, 124:187–192, 2002.

[30] D. Lakehal, G. S. Theodoris, and W. Rodi. Three-dimensional flow and heat tranfer calculations of film cooling at the leading edge of a symmetrical turbine blade model. *International journal of heat and fluid flow*, 22: 113–122, 2001.

[31] Azzi A. and Jubran B., A. Numerical modeling of film cooling from short length stream-wise injection holes. *Heat and Mass Transfer*, 39:345–353, 2003.

[32] Lakehal D., Theodoris G., S., and Rodi W. Computation of film cooling of a flat plate by lateral injection from arow of holes. *International Journal of Heat and Fluid Flow*, 19: 418–430, 1998.

[33] Bergeles, G. Gosman A. D., and Launder. The turbulent jet in a cross stream at low injection rates: a three-dimensional numerical treatment. *Numerical Heat Transfer*, 1:217–242, 1978.

[34] A. Azzi and D. Lakehal. Perspectives in modeling film cooling of turbine blades by transcending conventional two-equation turbulence models. *Journal of turbomachinery*, 124:472–484, 2002.

[35] Menter F., Thomas E., and Wolfgang V. Heat transfer predictions based on two-equation turbulence models. *ASME-JSME 2003 Thermal Engineering Joint Conference*, 16-20/03/2003.

[36] Kader B., A. Temperature and concentration profiles in fully turbulent boundary layers. *International Journal of Heat and Mass Transfer*, 24(2):1541–1544, 1982.

[37] A. Andreini, C. Bianchini, B. Facchini, and L. Mangani. Development and validation of a c++ object oriented cfd code for heat transfer analysis. *ASME-JSME 2007 Thermal Engineering and Summer Heat Transfer Conference*, 08-12/07/2007.

[38] K. Y. Chien. Predictions of channel and boundary-layer flows with a low reynolds number turbulence model. *AIAA Journal*, 20(1):33–38, 1982.

[39] K. Wieghardt and W. Tillman. On the turbulent friction layer for rising pressure. Technical Report TM-1314, NACA, 1951.

[40] D. E. Coles and E. A. Hirst. Computation of turbulent boundary layers. In *AFOSRIFP Stanford Conference*, volume 2. Stanford University, 1969.

[41] V. C. Patel, W. Rodi, and G. Sheuerer. Turbulence models for near wall and low reynolds number flows: a review. *AIAA Journal*, 26:1308–1319, 1993.

[42] Incropera F., P. and DeWitt D., P. *Fundamentals of Heat and Mass Transfer, 5th Edition*.

[43] D. Cooper, D. C. Jackson, B. E. Launder, and G. X. Liao. Impinging jet studies for turbulence model assessment. part i: Flow-field experiments. *Int. J. Heat Mass Transfer*, (36): 2675–2684, 1993.

[44] P. A. Durbin and S. Parneix M. Benhia. Prediction of heat transfer in an axisymmetric turbulent jet impinging on a flat plate. *International journal of heat mass tranfer*, 41(12): 1845–1855, 1998.

[45] B. Facchini and M. Surace. Impingement cooling for modern combustors: experimental analysis of heat transfer and effectiveness. *Experiments in Fluids*, (40):601–611, 2006.

[46] A. Andreini, C. Carcasci, S. Gori, and M. Surace. Film cooling system numerical design: adiabatic and conjugate analysis. *ASME paper*, (HT2005-72042), 2005.

[47] A. Azzi and D. Lakehal. Perspectives in modeling film cooling of turbine blades by transcending conventional two-equation turbulence models. *Journal of turbomachinery*, (124):472–484, 2002.

[48] A. Azzi and B. A. Jubran. Numerical modeling of film cooling from short length steam-wise injection holes. *Heat and mass transfer*, (39):345–353, 2003.

[49] D. Lakehal. Near-wall modeling of turbulent convective heat transport in film cooling of turbine blades with the aid of direct numerical simulation data. *Journal of turbomachinery*, (124):485–498, 2002.

[50] A. K. Sinha, D. G. Bogard, and M. E. Crawford. Film-cooling effectiveness downstream of a single row of holes with variable density ratio. *ASME Journal of Turbomachinery*, 113:442–449, 1991.

[51] L. Arcangeli, M. Surace, L. Tarchi, D. Coutandin, and S. Zecchi. Correlative analysis of effusion cooling systems. *ASME Turbo Expo*, (GT2006-90405), 2006.